

Using GD&T Tools in AutoCAD

AutoCAD allows you to add GD&T symbols to drawings using the **TOLERANCE** command. Accessing the **TOLERANCE** command displays the **Geometric Tolerance** dialog box. This is the primary method for adding feature control frames, geometric tolerancing symbols, and datum feature symbols. You can connect a leader to GD&T symbols using the **QLEADER** or **MLEADER** command. The **QLEADER** command provides a quick and effective option for attaching GD&T symbols to a leader.

Another option for placing GD&T symbols is to create your own blocks with attributes. You can insert blocks into the drawing and adjust the attribute data as needed. You can also add blocks to multileader lines using the **Block** multileader content type. Draw GD&T symbols on a dimensioning layer so the symbols and text can plot as objects that have the same thickness as extension and dimension lines (.01" or 0.3 mm). The suggested text font is romans.shx. These practices correspond with the ASME Y14.2M standard, *Line Conventions and Lettering*.

Using the **TOLERANCE** Command

Access the **TOLERANCE** command to open the **Geometric Tolerance** dialog box, which provides options for creating feature control frames. See **Figure 19A-1**. Areas divide the **Geometric Tolerance** dialog box into groups of compartments that relate to the compartments found in a feature control frame. Each area contains two levels to define a feature control frame.

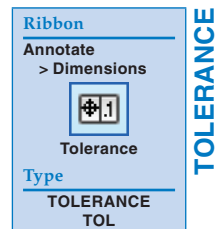
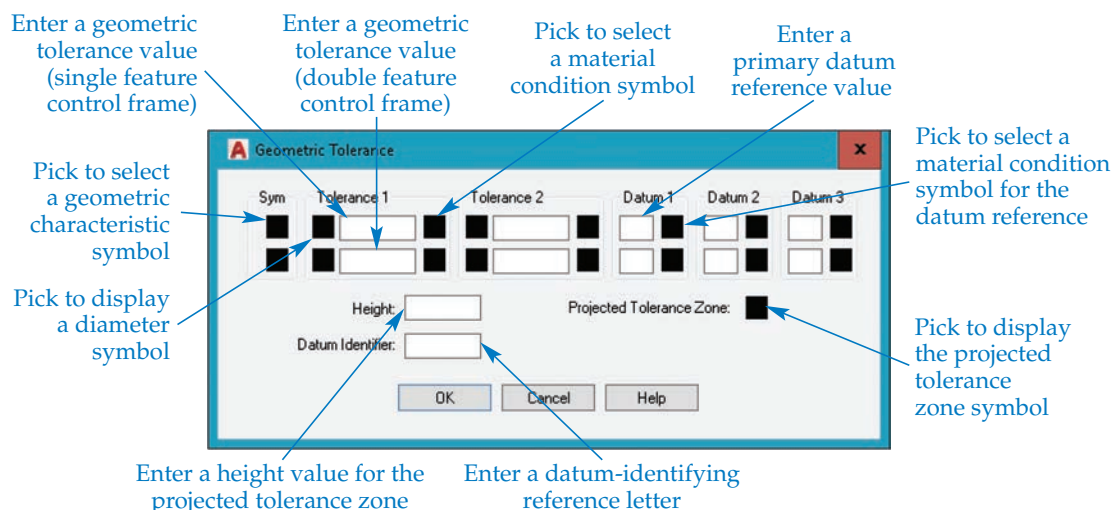


Figure 19A-1.

You can use the **Geometric Tolerance** dialog box to draw geometric dimensioning and tolerancing (GD&T) symbols and feature control frames.



The first, or upper, level allows you to create a single feature control frame. The lower level allows you to create a double feature control frame. The dialog box also provides options for displaying a diameter symbol and a modifying symbol. In addition, the **Geometric Tolerance** dialog box allows you to display a projected tolerance zone symbol and value and create a datum identifier for a datum feature symbol.

Selecting a Geometric Characteristic Symbol

You can access geometric characteristic symbols from the **Sym** area located at the far left of the **Geometric Tolerance** dialog box. This area has two boxes that allow you to display one or two geometric characteristic symbols. Pick one of the boxes in the **Sym** area to display the **Symbol** dialog box. See [Figure 19A-2](#). Pick a symbol to add the symbol to the selected **Sym** box. After selecting, the **Geometric Tolerance** dialog box returns. Pick the same box again to select a different symbol if necessary. To remove a symbol, access the **Symbol** dialog box and pick the blank image in the lower-right corner.

Tolerance 1 Area

The **Tolerance 1** area allows you to enter the first geometric tolerance value applied to the feature control frame. If drawing a single feature control frame, enter the desired value in the upper text box. If drawing a double feature control frame, also enter a value in the lower text box. You can add a diameter symbol by picking the box to the left of the text box. Pick the diameter box again to remove the diameter symbol.

The box to the right of the text box adds a material condition symbol. Pick the box to display the **Material Condition** dialog box shown in [Figure 19A-3](#). Pick the desired symbol to display it in the box you selected. To remove a material condition symbol, pick the blank tile in the **Material Condition** dialog box. [Figure 19A-3](#) shows the maximum material condition (MMC) and least material condition (LMC) symbols. The ANSI Y14.5M-1982 standard uses the regardless of feature size (RFS) symbol, but ASME Y14.5-2009 does not, because RFS is assumed unless otherwise specified.

In [Figure 19A-4](#), the **Sym** image tile includes a position symbol, and a 0.5 tolerance value is entered in the upper text box in the **Tolerance 1** area. A diameter symbol precedes the tolerance value, and the MMC symbol follows. Remember that a zero precedes metric decimals, but not inch decimals.

Figure 19A-2.

Use the **Symbol** dialog box to select a geometric characteristic symbol for use in a feature control frame.



Figure 19A-3.

The **Material Condition** dialog box. Notice that the symbol for regardless of feature size (RFS) is available. ASME Y14.5-2009 does not use this symbol, but you may need it when editing older drawings.

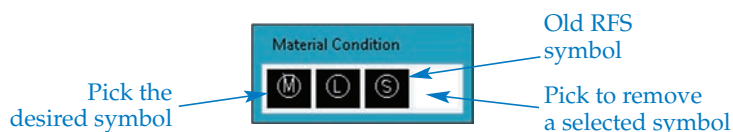
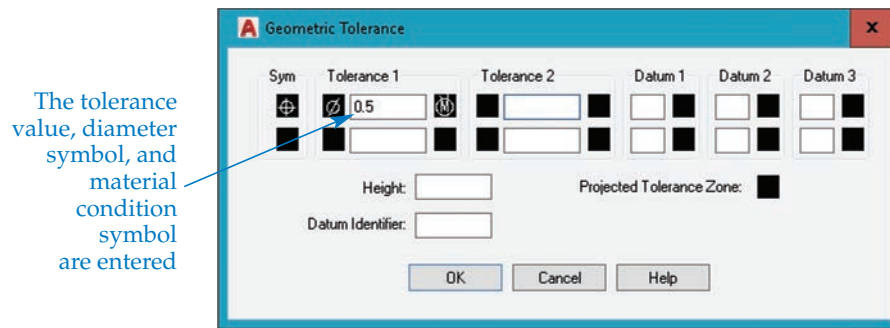


Figure 19A-4.

The **Geometric Tolerance** dialog box with a diameter symbol, geometric tolerance value, and maximum material condition (MMC) symbol added to the **Tolerance 1** area.



Tolerance 2 Area

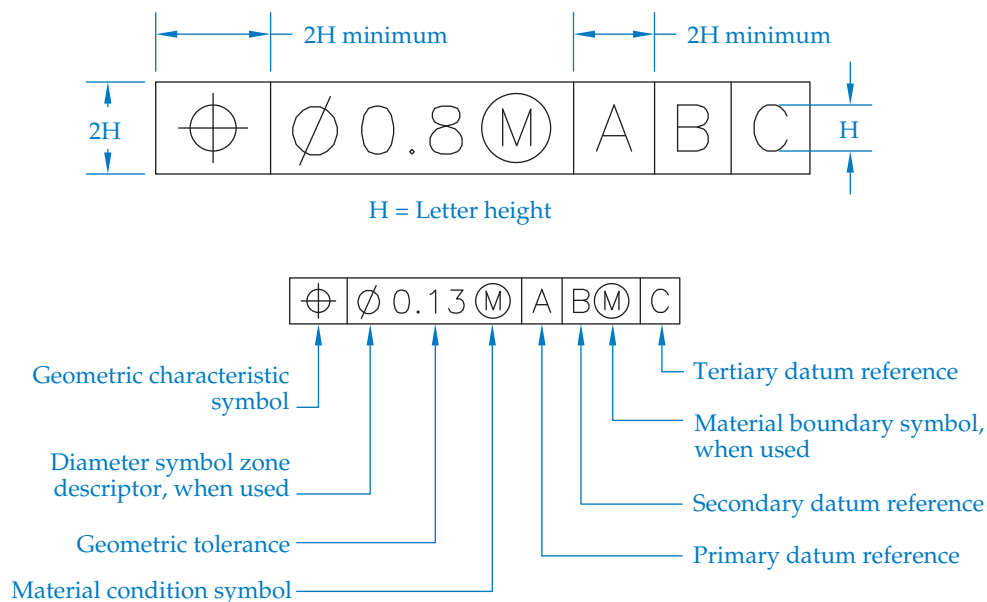
The **Tolerance 2** area allows you to add a second geometric tolerance to the feature control frame. This is not a common application, but is appropriate in some cases when there are restrictions on the geometric tolerance specified in the first compartment. For example, a second geometric tolerance value of 0.8 MAX maintains the specification given in the first compartment, but indicates that it cannot exceed 0.8.

Datum Areas

The **Datum 1** area establishes the information needed for the primary datum reference compartment. Like the **Tolerance** areas, this area offers two levels of text boxes to create single or double feature control frames. You can also specify a material condition symbol for the datum reference by picking the box to the right of the corresponding text box to open the **Material Condition** dialog box. The **Datum 2** and **Datum 3** areas allow you to specify the secondary and tertiary datum reference information. Refer to **Figure 19A-5** to see how the datum reference and related material condition symbols appear in the feature control frame.

Figure 19A-5.

The order of elements in a feature control frame.



Projected Tolerance Zone Box and Height Text Box

You can pick the **Projected Tolerance Zone:** box to display a projected tolerance zone symbol in the feature control frame. The **Height:** text box specifies the height of the projected tolerance zone. Note that the resulting display of the projected tolerance zone symbol by AutoCAD does not comply with ASME Y14.5-2009. Modifying a feature control frame to display the symbol correctly is discussed in the *Specifying a Projected Tolerance Zone* section.

Datum Identifier Text Box

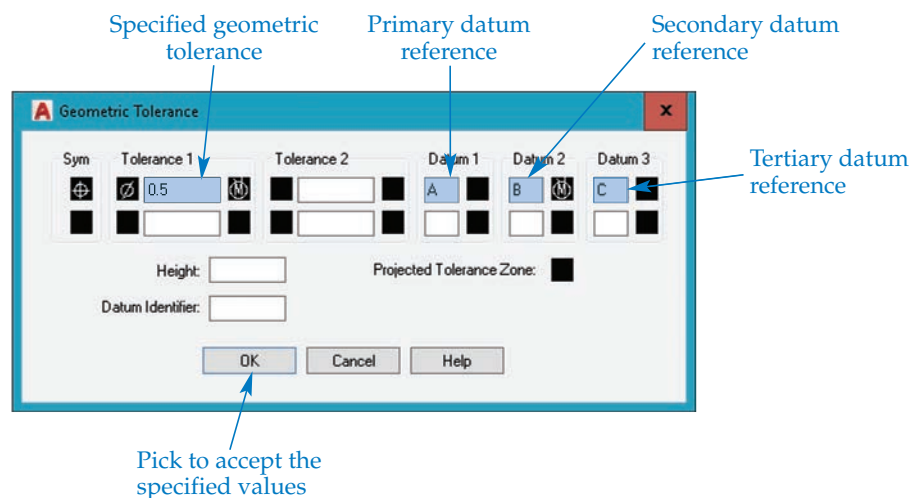
The **Datum Identifier:** text box is used to enter a datum-identifying reference letter as an element of a datum feature symbol. Use an uppercase letter. Creating a datum feature symbol with a connecting leader is discussed in the *Drawing Datum Feature Symbols* section.

Completing the Tolerance Command

After you enter the required information in the **Geometric Tolerance** dialog box, pick the **OK** button and pick a point to place the symbol in the drawing. **Figure 19A-6** shows the feature control frame for the given example. The height of the feature control frame is automatically set to twice the height of the text. Text on engineering drawings is generally .12" (3 mm) tall, which makes the feature control frame height .24" (6 mm). This complies with the ASME Y14.5 standard.

Figure 19A-6.

This example shows primary, secondary, and tertiary datum references added and highlighted, along with the geometric tolerance value. The resulting feature control frame appears below.



Feature Control Frame

Attaching Feature Control Frames to Leaders

In many cases, a feature control frame connects to a leader. The **QLEADER** command allows you to draw leader lines and access the **Geometric Tolerance** dialog box used to create feature control frames in one operation. This is the most effective technique for creating a feature control frame automatically attached and associated with a leader. You can create other GD&T symbols, such as datum feature symbols, more effectively using different methods.

Using the QLEADER Command

Access the **QLEADER** command to place a leader and attach a feature control frame in one operation. Dimension style settings control some of the leader line characteristics, such as the arrowhead size. The **Settings** option of the **QLEADER** command controls other elements, such as the leader format and annotation style.

Type
QLEADER
LE

QLEADER

NOTE

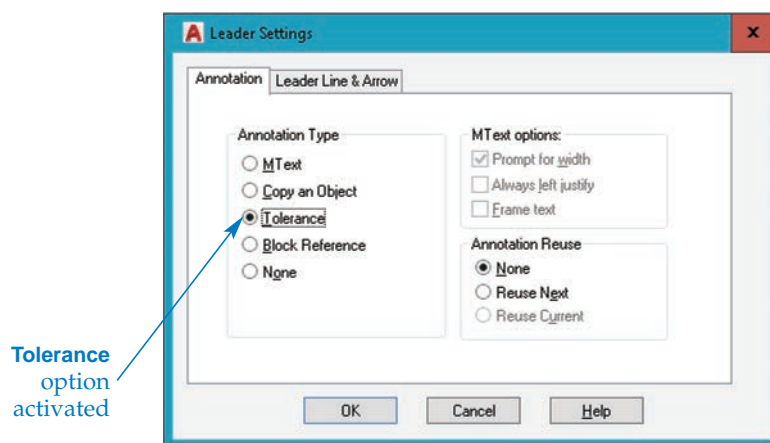
You can also use the **LEADER** command to automatically attach GD&T symbols to leaders. However, this command does not provide the same convenience and ability to comply with drafting standards as the **QLEADER** command.

When you enter the **QLEADER** command, use the **Settings** option to display the **Leader Settings** dialog box, [Figure 19A-7](#). In the **Annotation** tab, pick the **Tolerance** radio button to display the **Geometric Tolerance** dialog box for creation of a feature control frame with the leader line.

Next, select the **Leader Line & Arrow** tab of the **Leader Settings** dialog box. Pick the **Straight** radio button to create a leader with straight-line segments. When adding a feature control frame to a leader line, you should set the maximum number of vertices in the **Maximum** text box of the **Number of Points** area to 2. When you set the maximum number of leader points to 2, you select the start and endpoints of the leader line. Then the **QLEADER** command stops drawing the leader, automatically places the leader shoulder, and displays the **Geometric Tolerance** dialog box.

Figure 19A-7.

The **Leader Settings** dialog box. Activate the **Tolerance** radio button to place a feature control frame.



The **Arrowhead** area of the **Leader Line & Arrow** tab uses the default value assigned to leaders within the current dimension style. To change the appearance of the arrowhead, pick the drop-down list and select a terminator from the full range of choices.

You can restrict the first two segments of the leader line to certain angles using options in the **Angle Constraints** area of the **Leader Line & Arrow** tab. The options for each segment are **Any angle**, **Horizontal**, **90°**, **45°**, **30°**, and **15°**. The current **Ortho** mode setting in AutoCAD overrides the angle constraints, so it is advisable to turn **Ortho** mode off while using this command.

Pick the **OK** button to exit the **Leader Settings** dialog box. When asked to specify the first leader point, pick the location where the arrowhead points. Then pick the end of the leader line. If the maximum number of leader points is set to 2, the **Geometric Tolerance** dialog box displays. Otherwise, press [Enter] to end the leader line and display the **Geometric Tolerance** dialog box. Specify the settings and values for the feature control frame, and pick the **OK** button. The feature control frame connects to the leader line, as shown in **Figure 19A-8**.

Using the MLEADER Command

You can use the **MLEADER** command to create leaders, but it does not have an option to create a feature control frame at the same time. As a result, you must draw the leader separately using the **MLEADER** command and draw the feature control frame using the **TOLERANCE** command. Apply the **None** multileader content type when using this method. You can draw the leader before or after drawing the symbol. See **Figure 19A-9**.

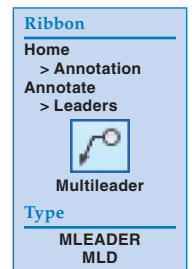


Figure 19A-8.

When you complete the **QLEADER** command, the feature control frame connects to the leader line.

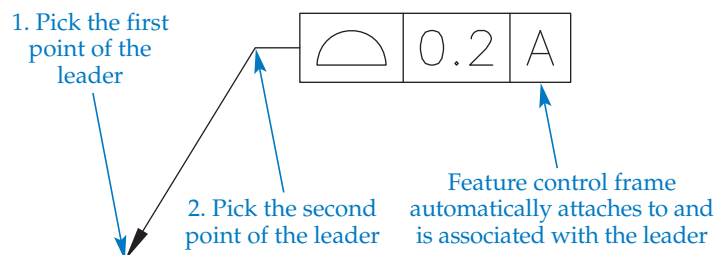
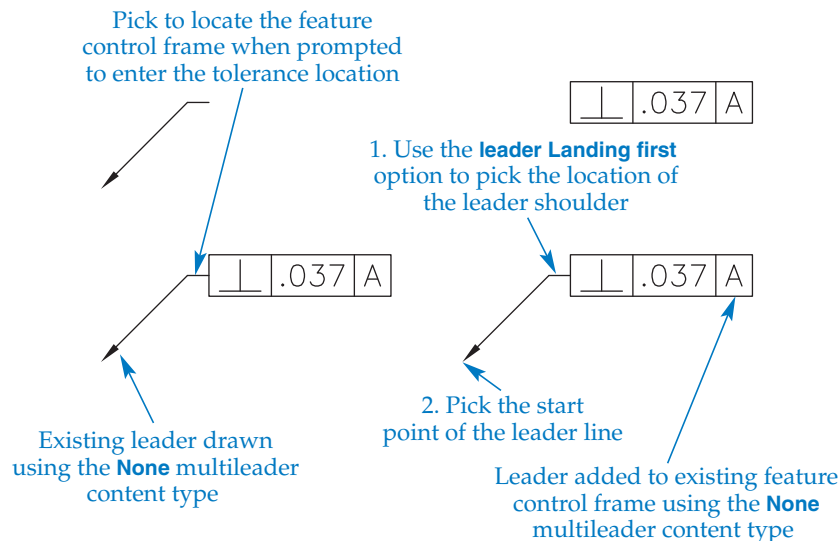


Figure 19A-9.

Use the **MLEADER** command to create a leader before drawing the feature control frame using the **TOLERANCE** command, or add the leader to an existing feature control frame.



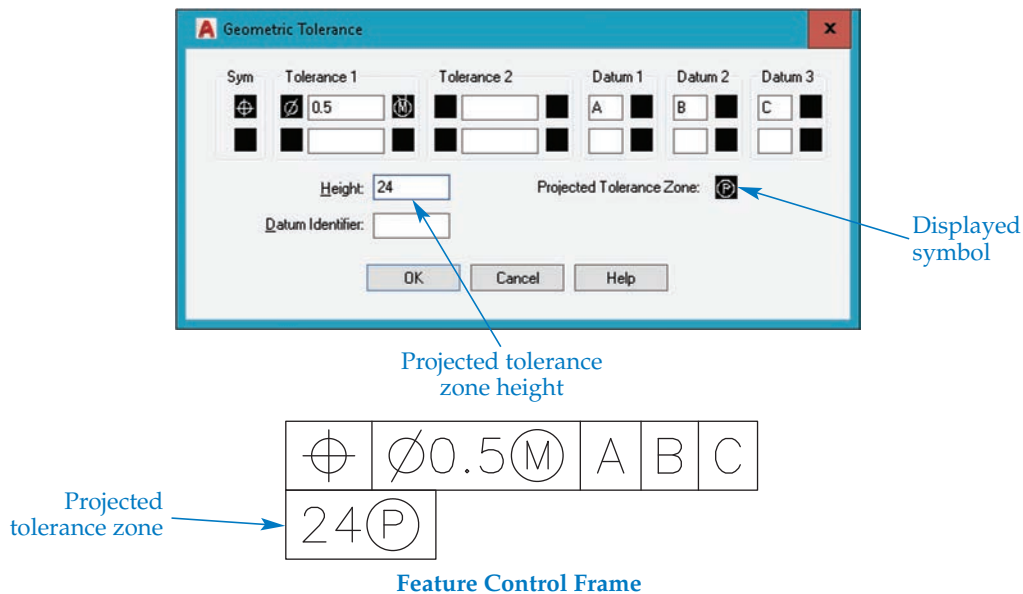
Specifying a Projected Tolerance Zone

AutoCAD specifies projected tolerance zones according to the ANSI Y14.5M-1982 standard. When following this standard, enter the desired geometric tolerance, diameter symbol, material condition symbol, and datum reference in the **Geometric Tolerance** dialog box, as previously described. Pick the **Projected Tolerance Zone:** box to display the projected tolerance zone symbol and enter the height in the **Height:** text box. See [Figure 19A-10](#). Place the feature control frame in the desired location in the drawing. Notice that AutoCAD displays the projected tolerance zone height in a separate compartment below the feature control frame, in accordance with ANSI Y14.5M-1982.

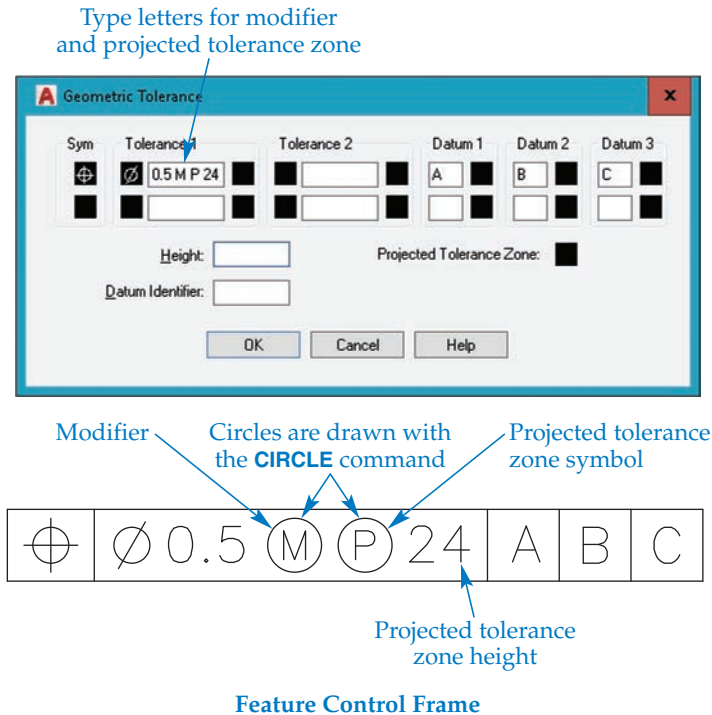
To specify a projected tolerance zone according to ASME Y14.5-2009, create a feature control frame with any modifier letters and the letter P after the tolerance value. Type the height of the projected tolerance zone after the P, and leave one space between each letter and the height value. See [Figure 19A-11](#). Then use the **CIRCLE** command to draw a circle around the modifier and the letter P. You can use the **BLOCK** command to create a block of the feature control frame and circles. This saves the symbol as a block, which can then be inserted as needed for repeated applications requiring the symbol. Creating a block “groups” the objects making up the block so that they are selectable as a single object.

Figure 19A-10.

To add projected tolerance zone specifications in accordance with ANSI Y14.5M-1982, enter the projected tolerance zone height and symbol in the **Geometric Tolerance** dialog box.



Specifying a projected tolerance zone in accordance with ASME Y14.5-2009.



Drawing a Double Feature Control Frame

Several GD&T applications require that you double the feature control frame in height, with two sets of geometric tolerancing values provided. These applications include unit straightness and flatness, composite positional tolerancing, and coaxial positional tolerancing. To draw a double feature control frame, use the **TOLERANCE** command to create the first level of the feature control frame in the **Geometric Tolerance** dialog box as previously described. You can also use the **QLEADER** command if connecting the feature control frame to a leader line. Then, pick the lower box in the **Sym** area. When the **Symbol** dialog box appears again, pick another geometric characteristic symbol. This results in two symbols in the **Sym** area. Continue specifying the needed information in the lower-level **Tolerance** and **Datum** text boxes. See [Figure 19A-12](#).

A composite frame forms when the symbols in the two **Sym** boxes are the same. Refer to [Figure 19A-12](#). Some situations require the same geometric characteristic symbol twice, one in the upper frame and another in the lower frame. To create two single-segment feature control frames, draw two separate feature control frames and create a block from them. If you are drawing a double feature control frame with different geometric characteristic symbols for a combination control, the feature control frame must have two separate compartments. See [Figure 19A-13](#).

Figure 19A-12.

Specifying information for a double feature control frame in the **Geometric Tolerance** dialog box.

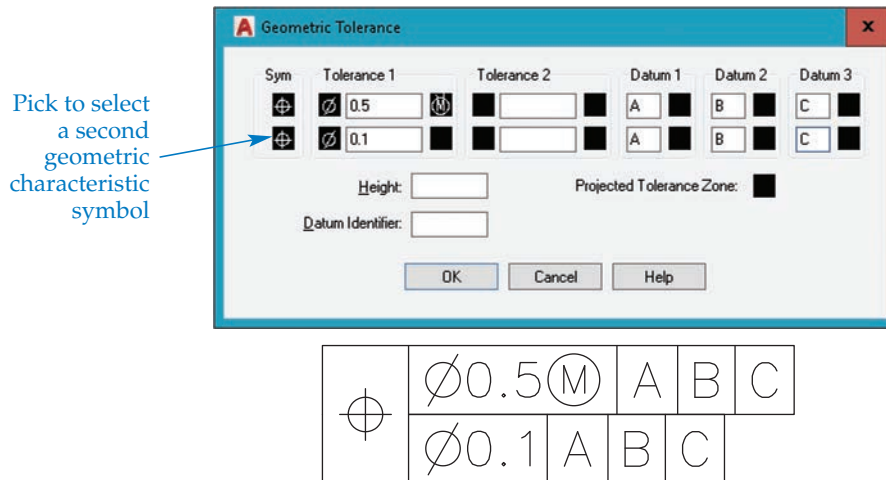
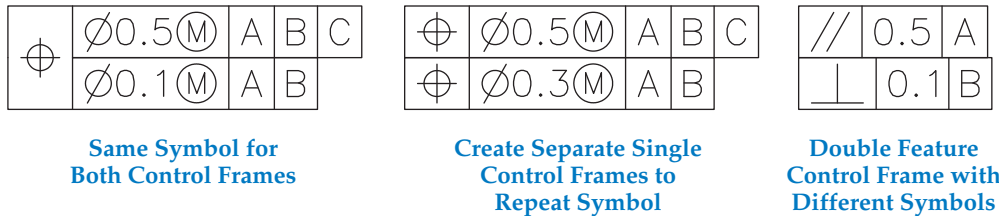


Figure 19A-13.

If you enter the same geometric characteristic symbol in both **Sym** boxes of the **Geometric Tolerance** dialog box, only one symbol appears in the first compartment of the feature control frame. Create two separate feature control frames to display the same symbol in both frames. If you use two different symbols, they appear in separate compartments.



Drawing Datum Feature Symbols

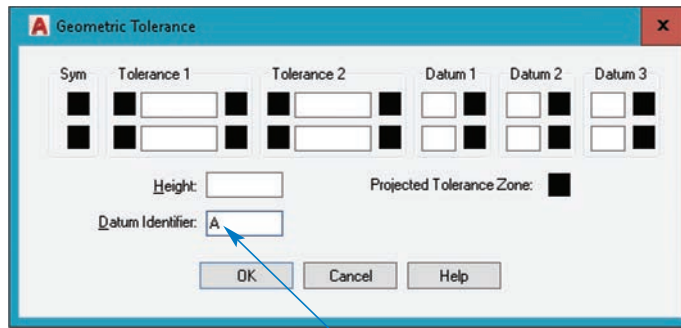
You can draw datum feature symbols using the **TOLERANCE** and **MLEADER** or **QLEADER** commands. Usually, you must use a combination of the **TOLERANCE** and **MLEADER** or **QLEADER** commands to draw an appropriate datum feature symbol. The method used to draw a datum feature symbol depends on the feature the symbol identifies. When you use the **Geometric Tolerance** dialog box to specify a datum feature symbol, enter the datum reference letter in the **Datum Identifier:** text box. See **Figure 19A-14**.

Options for Drawing Datum Feature Symbols

The datum feature symbols shown in **Figure 19A-15** are drawn using the **TOLERANCE** and **MLEADER** or **QLEADER** commands. One option is to use the **TOLERANCE** command first to place the datum identifier and then add a leader that connects the feature to the identifier. The other option is to draw a leader first and then use the **TOLERANCE** command to add the datum identifier. This usually requires you to move the datum identifier to the correct location using object snaps. **Figure 19A-16** shows both methods.

Figure 19A-14.

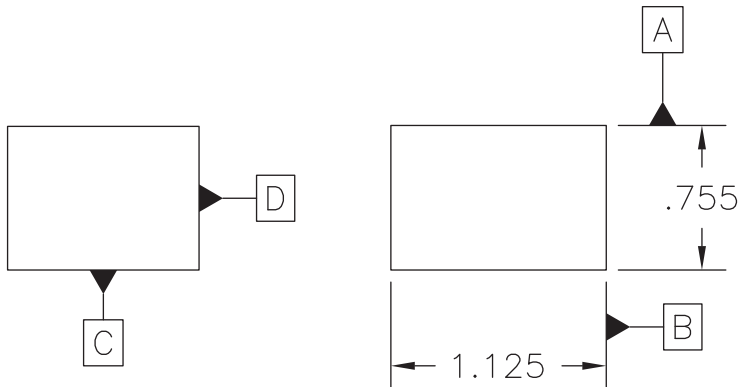
Using the **Geometric Tolerance** dialog box to enter a datum-identifying reference letter. The letter creates the datum identifier.



Specified datum
reference letter

Figure 19A-15.

Examples of datum feature symbols created using a combination of the **TOLERANCE** and **MLEADER** or **QLEADER** commands.



When you use the **MLEADER** command to add the leader, create a separate multi-leader style with the **Datum triangle filled** arrowhead symbol, set the maximum leader points to 2, do not include a landing, and use the **None** multileader content type. When you use the **QLEADER** command to add the leader, create a dimension style that uses the **Datum triangle filled** arrowhead symbol, use the **None** annotation type, and set the maximum leader points to 2. When a datum feature symbol requires a shoulder, add the shoulder manually by picking a third point. This avoids shifting the angle of the leader line.

Adding Datum Feature Symbols to Angled Surfaces

You must follow specific steps in order to add a datum feature symbol to an angled surface, as shown in **Figure 19A-17**. One option is to use the **QLEADER** command. Before adding the leader, create a dimension style that uses the **Datum triangle filled** arrowhead symbol. Then enter the **QLEADER** command and use the **Settings** option to open the **Leader Settings** dialog box. Select the **Annotation** tab and pick the **Tolerance** radio button. Select the **Leader Line & Arrow** tab of the **Leader Settings** dialog box and pick the **Straight** radio button. When adding a datum feature symbol to the leader line, you should set the maximum number of vertices in the **Maximum** text box of the **Number of Points** area to 3. This allows you to construct the leader shoulder manually. If you let AutoCAD form the leader shoulder automatically, it shifts the angle of the leader line.

Figure 19A-16.

Use the **MLEADER** or **QLEADER** command to add a leader before drawing a datum identifier, or add the leader to an existing datum identifier.

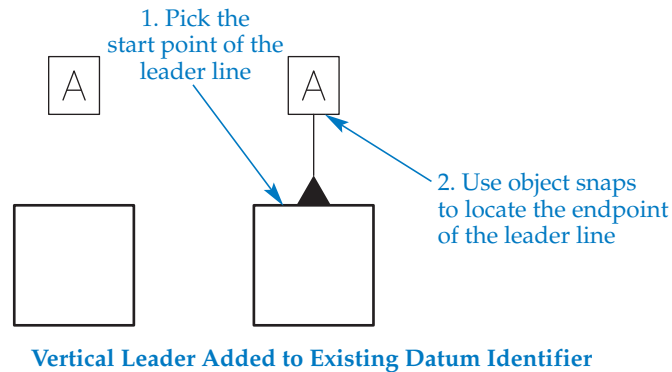
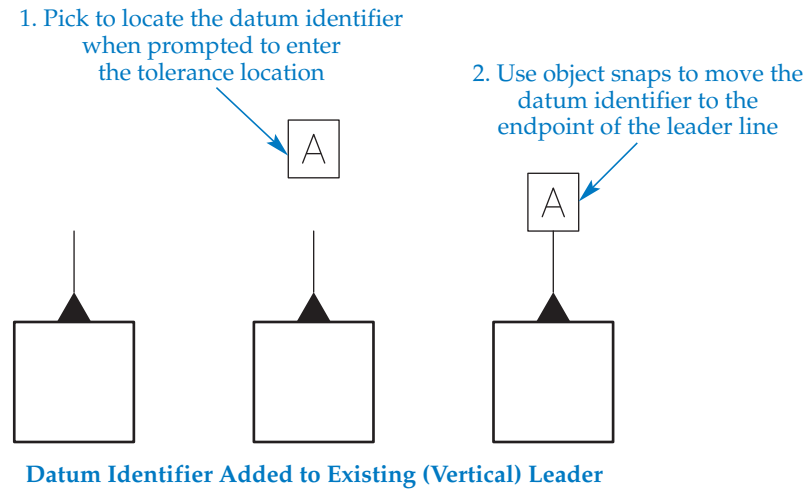
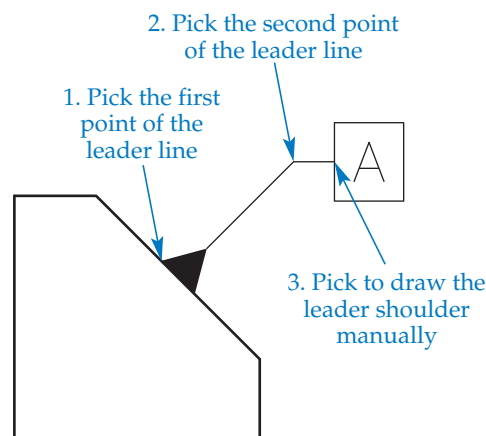


Figure 19A-17.

Use the **Tolerance** annotation option of the **QLEADER** command to add a datum feature symbol to an angled surface.



Pick the **OK** button to exit the **Leader Settings** dialog box. Pick the leader start point and then the next leader point. The second point must create a line segment that is perpendicular to the angled surface. Pick the third point to define the length of the leader shoulder. If the maximum number of leader points is set to 3, the **Geometric Tolerance** dialog box displays. Otherwise, press [Enter] to end the leader line and display the **Geometric Tolerance** dialog box. Specify a value in the **Datum Identifier**: text box and pick the **OK** button.

Drawing Basic Dimensions

Figure 19A-18 shows a basic dimension. You can draw basic dimensions automatically by setting a basic tolerance in the **Tolerances** tab of the **New** (or **Modify**) **Dimension Style** dialog box. Typically, you establish a separate dimension style for basic dimensions because not all of the dimensions on a drawing are basic.

The height of the basic dimension rectangle is twice the height of the text, as shown in **Figure 19A-19**. Text on engineering drawings is generally .12" (3 mm) tall, which makes the basic dimension rectangle height .24" (6 mm). As a result, the distance from the text to the basic dimension rectangle should be equal to half the text height. For example, if the height of the drawing text is .12", the space between the text and the basic dimension rectangle should be .06" to result in a .24" high frame. The **Offset from dim line**: setting in the **Text** tab of the **New** (or **Modify**) **Dimension Style** dialog box controls the distance from the text to the basic dimension rectangle. The setting also controls the gap between the dimension line and the dimension text for linear dimensions. Picking the **Draw frame around text** check box in the **Text** tab of the **New** (or **Modify**) **Dimension Style** dialog box also activates the basic tolerance method.

Figure 19A-18.
A basic dimension.

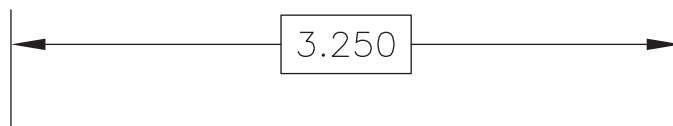
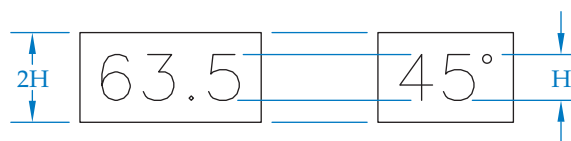


Figure 19A-19.
The height of the rectangle drawn around basic dimension text is twice the text height by default.



H = Letter height



The number of times or places can be applied to a basic dimension by placement inside or outside of the basic dimension symbol.

Editing Feature Control Frames

A feature control frame created using the **TOLERANCE** command is an AutoCAD tolerance object. Edit feature control frames using editing commands such as **ERASE**, **COPY**, **MOVE**, **ROTATE**, and **SCALE**. The **STRETCH** command only allows you to move a feature control frame. This effect is similar to the results of using the **STRETCH** command with text objects.

Double-click on a tolerance object or use the **TEXTEDIT** command to reenter the **Geometric Tolerance** dialog box with the values assigned to the feature control frame. After you make changes, pick **OK** to update the feature control frame.

You can also use the **TEXTEDIT** command to edit basic dimensions. Select the basic dimension for editing to display the text editor. You can then edit the basic dimension as you would any other dimension.

Type
TEXTEDIT TEDIT

TEXTEDIT